Compatibility Notes

These notes give an overview of changes between versions 3.3 and 3.4 of COMSOL Multiphysics that require special attention with regards to backward compatibility.

Renamed Dialog Box

The **ODE Settings** dialog box in version 3.3 has been renamed **Global Equations** in version 3.4 to better reflect the range and versatility of the functionality provided.

Boundary Conditions for Fluid Flow

The boundary conditions for the following application modes have been upgraded:

- Incompressible Navier-Stokes in COMSOL Multiphysics and the Chemical Engineering Module, Earth Science Module, and MEMS Module
- General Laminar Flow in the MEMS Module
- Non-Newtonian Flow in the Chemical Engineering Module
- Weakly Compressible Navier-Stokes in the Chemical Engineering Module, Heat Transfer Module, and MEMS Module
- Stokes Flow in the MEMS Module
- Brinkman Equations in the Chemical Engineering Module and Earth Science Module
- k-ε Turbulence Model and k-ω Turbulence Model in the Chemical Engineering Module and Heat Transfer Module
- Level Set Two-Phase Flow in the Chemical Engineering Module and MEMS Module

The reason for this upgrade is two-fold:

- to make the modeling process more user friendly
- to provide greater stability for solutions of larger problems when using the iterative solvers

The primary impact of these new boundary conditions is found in pressure conditions at outlets. The previous pressure condition is automatically replaced with the mathematically equivalent condition, which in version 3.4 is denoted **Normal stress**.

You can obtain greater stability with the new **Outlet** type boundary condition **Pressure, no viscous stress**, which is now the default condition on outlets.

The following lists provide information about how COMSOL Multiphysics converts fluid-flow boundary conditions in version 3.3 to the corresponding conditions in version 3.4.

TOTAL STRESS TENSOR, NONTURBULENT FLOWS

- **Velocity**: Translates to **Boundary type**: **Inlet**, **Boundary condition**: **Velocity**.
- **Pressure**: Translates to **Boundary type**: **Outlet**, **Boundary condition**: **Normal stress**.
- **Slip/Symmetry**: Translates to **Boundary type**: **Symmetry boundary**.
- **No slip**: Translates to **Boundary type**: **Wall**, **Boundary condition**: **No slip**.
- **Normal flow, pressure**: Translates to **Boundary type**: **Stress**, **Boundary condition**: **Normal stress, normal flow**.
- **Neutral**: Translates to **Boundary type**: **Open boundary**, **Boundary condition**: **Normal stress**.
- **Laminar inflow/outflow**: Translates to **Boundary type**: **Inlet**, **Boundary condition**: **Laminar inflow**.

VISCOUS STRESS TENSOR, NONTURBULENT FLOWS

- **Velocity**: Translates to **Boundary type**: **Inlet**, **Boundary condition**: **Velocity**.
- **Pressure**: Translates to **Boundary type**: **Outlet**, **Boundary condition**: **Pressure, no viscous stress**.
- **Slip/Symmetry**: Translates to **Boundary type**: **Symmetry boundary**.
- **No slip**: Translates to **Boundary type**: **Wall**, **Boundary condition**: **No slip**.
- **Normal flow, pressure**: Translates to **Boundary type**: **Outlet**, **Boundary condition**: **Pressure, no viscous stress** (the old boundary condition was over specified and no mathematically equivalent counterpart can be formulated).
- **Neutral**: Translates to **Boundary type**: **Open boundary**, **Boundary condition**: **No viscous stress**.
- **Laminar inflow/outflow**: Translates to **Boundary type**: **Inlet**, **Boundary condition**: **Laminar inflow**.

TOTAL STRESS TENSOR, TURBULENT FLOWS

- **Inflow**: Translates to **Boundary type**: **Inlet**, **Boundary condition**: **Velocity**.
- **Outflow:** Translates to **Boundary type**: **Outlet**, **Boundary condition**: **Velocity**.

- **Pressure**: Translates to **Boundary type**: **Outlet**, **Boundary condition**: **Normal stress**.
- **Slip/Symmetry**: Translates to **Boundary type**: **Symmetry boundary**.
- **Normal flow, pressure**: Translates to **Boundary type**: **Stress**, **Boundary condition**: **Normal stress, normal flow**, with **Boundary type**: **Outlet** for the turbulence variables.
- **Neutral**: Translates to **Boundary type**: **Open boundary**, **Boundary condition**: **Normal stress**.
- **Logarithmic wall function**: Translates to **Boundary type**: **Wall**, **Boundary condition**: **Logarithmic wall function**.
- **Logarithmic moving wall**: Translates to **Boundary type**: **Wall**, **Boundary condition**: **Logarithmic moving wall**.

VISCOUS STRESS TENSOR, TURBULENT FLOWS

- **Inflow**: Translates to **Boundary type**: **Inlet**, **Boundary condition**: **Velocity**.
- **Outflow:** Translates to **Boundary type**: **Outlet**, **Boundary condition**: **Velocity**.
- **Pressure**: Translates to **Boundary type**: **Outlet**, **Boundary condition**: **Pressure, no viscous stress**.
- **Slip/Symmetry**: Translates to **Boundary type**: **Symmetry boundary**.
- **Normal flow, pressure**: Translates to **Boundary type**: **Outlet**, **Boundary condition**: **Pressure, no viscous stress**.
- **Neutral**: Translates to **Boundary type**: **Open boundary**, **Boundary condition**: **No viscous stress**.
- **Logarithmic wall function**: Translates to **Boundary type**: **Wall**, **Boundary condition**: **Logarithmic wall function**.
- **Logarithmic moving wall**: Translates to **Boundary type**: **Wall**, **Boundary condition**: **Logarithmic moving wall**.

NONTURBULENT FLOWS, INTERIOR BOUNDARIES

- **Pressure**: Translates to **Continuity**.
- **Slip/Symmetry**: Translates to **No slip**.
- **Normal flow, pressure**: Translates to **Continuity**.
- **Neutral**: Translates to **Continuity**.

TURBULENT FLOWS, INTERIOR BOUNDARIES

- **Outflow**: Translates to **Velocity**, with **Continuity** for the turbulence variables.
- All other conditions translate to **Continuity**.

LEVEL SET TWO-PHASE FLOW

- **Velocity**: Translates to **Boundary type**: **Inlet** or **Outlet**, depending on whether the boundary is an inflow or outflow boundary, **Boundary condition**: **Velocity**.
- **Pressure**: Translates to **Boundary type**: **Inlet** or **Outlet**, depending on whether the boundary is an inflow or outflow boundary, **Boundary condition**: **Pressure, no viscous stress**.
- **Normal flow/Pressure**: Translates to **Boundary type**: **Inlet** or **Outlet**, depending on whether the boundary is an inflow or outflow boundary, **Boundary condition**: **Pressure, no viscous stress**.
- **Neutral**: Translates to **Boundary type**: **Inlet**, **Boundary condition**: **Pressure, no viscous stress** with $\mathbf{p_0} = 0$.

The Turbulence Model Application Modes

The k-ε Turbulence Model and k-ω Turbulence Model application modes in the Chemical Engineering Module and Heat Transfer Module both make use of the logarithmic wall function boundary conditions.

In version 3.3, the logarithmic wall functions were implemented using Neumann conditions. These constraints adapt the flux, which in turn sets the value of the variables, that is, they do not set the value of the function explicitly.

In contrast, the strong formulation in version 3.4 sets the value of the variables by explicitly using the value of the logarithmic wall function. This yields a more accurate fulfillment of the boundary conditions and makes solutions of models that use iterative solvers significantly more robust than in version 3.3.

The change in the implementation of the logarithmic wall functions in version 3.4 can result in slightly different results compared to version 3.3. For example, the pressure loss in the Turbulent Backstep model in the Chemical Engineering Model Library differs by about 2% between versions 3.3 and 3.4.

The Convection and Conduction and General Heat Transfer Application Modes

The Convection and Conduction application modes in COMSOL Multiphysics and the Chemical Engineering Module as well as the General Heat Transfer application mode in the Heat Transfer Module have been upgraded to be compatible with weakly compressible flows. The equations are now formulated in a nonconservative form.

In addition, the Convection and Conduction application mode in the Chemical Engineering Module and the General Heat Transfer application mode in the Heat Transfer Module have received optional terms that account for pressure work and viscous heating. If you have manually included terms emanating from compressibility effects, you can most likely replace them using the new features for weakly compressible flow.

Predefined Couplings for Flow with Variable Density

The Non-Isothermal Flow application mode in version 3.3 of the Chemical Engineering Module, Heat Transfer Module, and MEMS Module is denoted Weakly Compressible Navier-Stokes in version 3.4.

In the Chemical Engineering Module version 3.4, Non-Isothermal Flow is a subgroup of the application mode Flow with Variable Density. It contains predefined multiphysics couplings for laminar and turbulent flows where the density depends on the temperature.

The Chemical Engineering Module version 3.3 had a group of predefined multiphysics couplings denoted Fluid-Thermal Interaction. In version 3.4, all functionality previously found under that heading is included in the predefined couplings in the Flow with Variable Density application mode.

The Predefined Coupling for Fluid-Structure Interaction

The default solver settings for fluid-structure interactions in version 3.4 of the MEMS Module and Structural Mechanics Module are greatly improved compared to version 3.3. Stationary analyses now use the new segregated solver, which yields a substantial decrease in memory consumption and solution time. This new solver has also made it possible to solve models an order of magnitude larger in terms of degrees of freedom compared to version 3.3.

DOF Numbering and Mesh Element Size Parameter Definition

The internal numberings of mesh vertices, mesh elements, and degrees of freedom have been changed between versions 3.3 and 3.4 to improve efficiency. Unless you explicitly access the elements of `fem.sol.u` in a model's FEM structure, these changes have no implications for backward compatibility.

In version 3.3, the mesh element size, h, was proportional to the square root of the area (2D) or cubic root of the volume (3D) of the mesh element. This measure is a good approximation as long as the mesh element is approximately isotropic. To get a better measure of the mesh element size for the anisotropic elements found in a boundary layer mesh, the definition of h has been changed in version 3.4 to the longest mesh element side. The old and the new measures are very similar for nearly isotropic mesh elements. The changed definition of h might have a small impact on models that rely heavily on the stability provided by artificial diffusion.